Analysis of 3-D Model in ANSYS 9.0 by Java Program and Macros Using Interlinking Concept Verification Though the CFD Analysis

S.S. Khedkar[1], S.S. Chaudhari[2],P.D.Kamble[3]
[1]Yeshwantrao Chavan College of Engineering, Nagpur, Maharashtra, India
sandip_khedkar@rediffmail.com
[2]Yeshwantrao Chavan College of Engineering, Nagpur, Maharashtra, India
sschaudharipatil@rediffmail.com
[3]Yeshwantrao Chavan College of Engineering, Nagpur, Maharashtra, India
pdk121180@yahoo.com

***Abstract* - ANSYS, one of the CAE software is widely used to perform various types of analysis under various loading condition. But the major drawback of the software is that it does not provide 'undo' option because of which it becomes quite hectic to edit the entered parameter in the same iteration. Also slight change in dimension or in boundary condition, force user to perform the same steps of analysis once again. Macro files are used to store ANSYS Parametric Design Language (APDL). So the objective of the present paper is to edit macro file as per the desired conditions which will reflect in ANSYS Parametric Design Language (APDL) program in ANSYS software. The main aim in this research work is to interlink macro file and computer program which will helpful to edit macro file conversely the updation will be seen in ANSYS model automatically. For detailed, CFD analysis of 2-D duct for laminar flow is considered. For different dimensions of 2-D duct, the model is tested and the loading condition is changed through a Java program. This work will be very useful to perform analysis for the beginner level user. Also the time required modifying the model and to perform its analysis is reduced promisingly in the area of CAE technology.**

***Index Terms* —ANSYS, Macro, APDL Programming, CFD Analysis, Laminar Flow, CAE**

I. INTRODUCTION

India is now moving towards golden era due to globalization of the market and the interest of so many companies to establish their plants recently. As far as modernization of industries is concerned, every industry has automatic machines, robots and sophisticated design centres. Whilst an accurate design of any product is a prime importance to an industry, this may includes, the design (on practice) of the products visualize and analyse by a CAD software and CAE software. It is in the domain of someone to graphically visualize the product for the given dimensions. ANSYS is a widely-used software tool for structural, thermal, and fluid flow analysis. The result obtained by the software is quite healthy, near to the accuracy and can be accepted. However, besides the advantages of the ANSYS it also have some drawbacks like significant amount of time is needed to learn to use a complex finite element package also there is no facility of 'undo' process to edit the dimensions entered previously in case of any wrong value entered.

Hence to correct the mistake user has to repeat the same scenario once again so as to get the result. Also if the result obtained is not satisfactory, the same scenario have to be replayed with change in the desired parameter until satisfactory result is obtained during design process. The repetition may include total process of creating model, loading of the model by various parameters like force, pressure, temperature, velocity, rotation etc and analysis of the same. In view of above stated problem our paper work presents parametric based design to put one approach for how to change parameters of APDL program through JAVA programming.

II. RESEARCH METHODOLOGY

Communicating with the ANSYS Program can be done via two ways

Graphical User Interface (GUI)

Command Prompt

The GUI consists of windows, menus, dialog boxes, and other components that allow us to enter input data and execute ANSYS functions simply by picking buttons with a mouse or typing in responses to prompts. Commands are the instructions that direct the ANSYS program. ANSYS has more than 1200 commands, each designed for a specific function. Most commands are associated with specific (one or more) processors, and work only with that processor or those processors. To use a function, it can be either typed in the appropriate command or the function can be accessed from the GUI (which internally issues the appropriate command). ANSYS commands have a specific format. A typical command consists of a command name in the first field, usually followed by a comma and several more fields (containing arguments). Frequently used sequence of ANSYS commands can be recorded in a macro file (with an extension of .MAC or .mac), thus creating a personalized ANSYS command. Execution of the same macro file executes the commands stored in it. Thus number of commands can be grouped into a single macro file and their sequential one by one execution can be reduced to the single execution of the macro file. APDL (ANSYS Parametric Design Language) is the format used by the macro file to store the ANSYS command. Hence the total research work can be divided into following main steps

DOI: 01.IJMMS.01.01.532

Development of the Macro file
Development of the JAVA code
Execution of the macro file
Development of the macro file:

The macro file stores the ANSYS commands which will used to, select the type of analysis, element type, material properties, creation of the model, meshing of the model, loading of the model, analysis of the model, and result review of the analyzed model either graphically or in tabulated format.

Development of JAVA Code:

The JAVA code is used to edit the values of various parameters stored in the macro file. The parameters may include physical dimension of the model, material properties of the model etc.

Execution of the macro file:

As the execution of the macro file in ANSYS is set up, the ANSYS will execute the commands stored in macro file and the analysis will be completed.

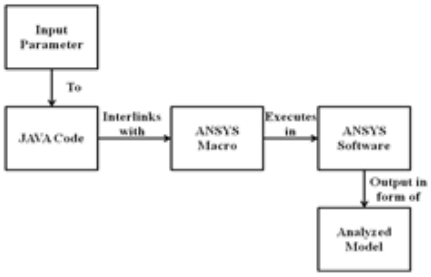


Figure1.Interlinking Process

III.EXPERIMENTATION

To illustrate the above mentioned steps briefly, the fluid flow problem of 'Steady State CFD Analysis of Laminar Air Flow In a 2-D Duct' can be considered for experimentation.

Problem Description

This problem models laminar air flow in a two-dimensional duct for convergent and divergent section individually. Following are various properties of duct and air.

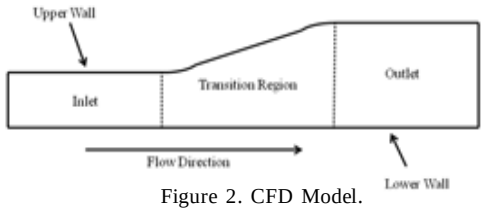


Figure 2. CFD Model.

TABLE 1. DIMENSIONS OF AIR FLOW DUCT.

Parameters	Convergent	Divergent
Inlet length	1.5 in	1.5 in
Inlet height	1.5 in	0.5 in
Transition length	5 in	5 in
Outlet height	0.5 in	1.5 in
Outlet length	2.5 in	2.5 in
Air density	1.21x10-7 lbf-s2/in4	1.21x10-7 lbf-s2/in4
Air viscosity	2.642x10-9 lbf-s/in2	2.642x10-9 lbf-s/in2
Inlet velocity	1 in/sec	1 in/sec
Outlet pressure	0 psi	0 psi

Now in this case, if user wants to perform CFD analysis for both convergent and divergent section. Separate one by one analysis has to be performed, though it has similarities. Methodology mentioned above will make it very simple and easy. By using the java interface developed for this particular application, the user has to enter the parameters which will change for both the sections. The JAVA code will edit the parameter and as the macro gets the executed, the desired result is obtained. Following are the detailed steps for this particular problem.

Development of the macro file:

The macro file is developed with the help of 'APDL programmers guide ANSYS 10.0' The macro will consist of commands which will select the type of analysis, element type, material properties, creation of the model, meshing of the model, loading of the model, analysis of the model, and result review of the analyzed model in graphical format. The scalar variable parameter has been set for the various values like inlet length, inlet height etc, which may change with the number of iterations.

Development of JAVA Code:

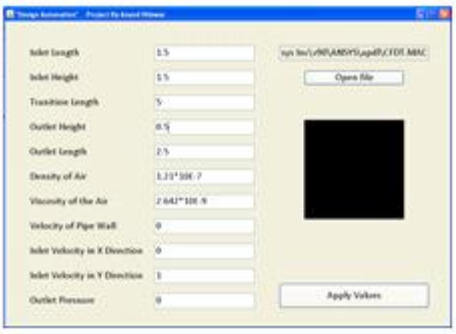
The JAVA code is used to edit the values of various variable parameters stored in the macro file. The parameters include Inlet length, Inlet height, Transition length, Outlet height, Outlet length, Air density, Air viscosity, Inlet velocity, Outlet pressure. By using the same JAVA application, number of times same scenario may be replayed with different parameters in each iteration by altering the value only in JAVA application. Hence the hectic procedure of ANSYS software is eliminated and time will be saved. The macro file is saved at location 'C:\Program Files\Ansys Inc\v90\ANSYS\apdl' which is default 'micro search path' In case of ANSYS 11.0 it will be 'C:\Program Files\Ansys Inc\v110\ANSYS\apdl'

Execution of the macro file:

To execute the macro in ANSYS, following sequence is used. Utility Menu→Macro→Execute Macro→Name of the macro to be executed= 'macro name'

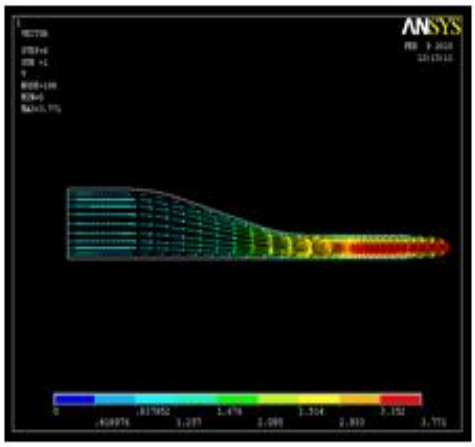
As a result of this, CFD analysis of the duct is completed and various results may be plot using various postprocessor commands through ANSYS.

Converaent Section

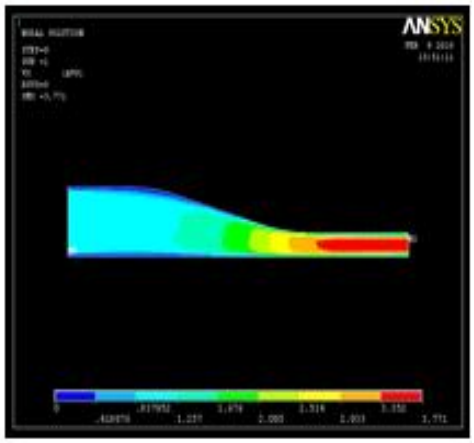


Picture1. Java interface for convergent section

DOI: 01.IJMMS.01.01.532



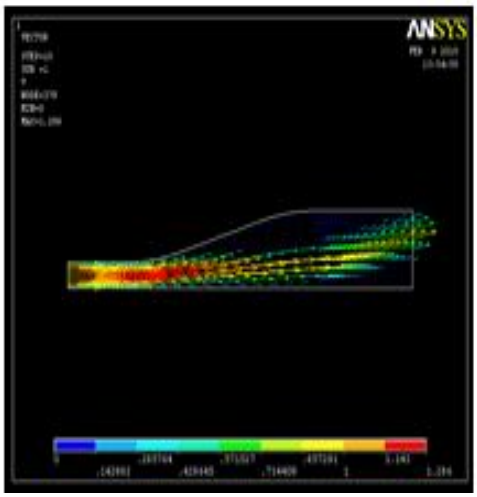
Picture2.Plot of velocity vector



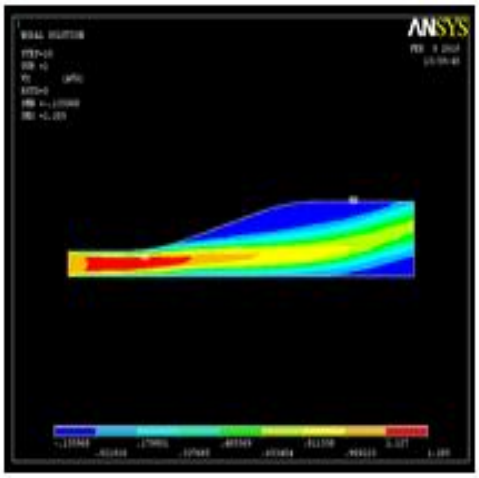
Picture 3.Plot of pressure distribution

Divergent Section

Picture 4. Java interface for divergent section



Picture 5. Plot of pressure distribution



Picture 6. Plot of pressure distribution

IV. CONCLUSION

Use of Macro file in ANSYS finite element software package has been overviewed. It seems that by altering the parameters in the macro file directly results in the alteration of the model in ANSYS software. System parameters can be varied and result can be observed as an effect on calculated system responses through high level graphical representations. Also the above mentioned work easily provides good supplementary information to reinforce concepts to the beginner level user of ANSYS. No FEA background is required on the part of the user or instructor. From the above mentioned experimental work, it can also be concluded that the efforts required in the repetitive analysis of same part can be reduced also time taken for analysis may be reduced to great extent.

V. REFERENCES

[1.] Andreas Hieke ,SIEMENS Microelectronics "Simple APDL Implementation Of A 3D FEM Simulation For Mutual Capacitances Of Arbitrarily Shaped Objects Like Interconnects", IBM Semiconductor Research & Development Center NY 12533, USA

[2.]S-Y. Chen, "Integrating ANSYS With Modern Numerical Optimization Technologies" ANSYS Solutions Magazine, Spring

DOI: 01.IJMMS.01.01.532

Issue Oct 2000,
[3.] John R. Baker "ANSYS Macros For Illustrating Concepts In Mechanical Engineering Courses" Department Of Mechanical Engineering University Of Kentucky
[4.] A Jeremy P. Dittmer1, C. Greg Jensen2, Michael Gottschalk3 and Thomas Almy4 "Mesh Optimization Using a Genetic Algorithm to Control Mesh Creation Parameters" .
[5.] Terrance W. Ewanochko1, David Nelson2, Martin Esaki2 Eric Pesheck3 "Integrated Durability Analysis Using ANSYS, ADAMS and FE"Fatigue".
[6.] Peter R. Barrett, Patrick Cunningham, "Super Elastic Alloy Eyeglass Frame Design Using the ANSYS Workbench Environment" *Computer Aided Engineering Associates Inc.*
[7.] Chandresh Shah, Cummins, Inc "Mesh Discretization Error and Criteria for Accuracy of Finite Element Solutions".
[8.] "APDL programmers guide ANSYS 10.0" by ANSYS, Inc.

DOI: 01.IJMMS.01.01.532